REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	June 2004	Technical Memorandum

4. TITLE AND SUBTITLE

Procedure for Determining One-Dimensional Flow Distributions in Arbitrarily Connected Passages Without the Influence of Pumping

5. FUNDING NUMBERS

WBS–22–722–90–C1
1L161102AF20

6. AUTHOR(S)

Peter L. Meitner

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER

E–13567

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001
and
U.S. Army Research Laboratory
Adelphi, Maryland 20783–1145

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM—2004-211885
ARL–TR–3213
GT2003–38061

11. SUPPLEMENTARY NOTES

Prepared for the Turbo Expo 2003 cosponsored by the American Society of Mechanical Engineers and the International Gas Turbine Institute, Atlanta, Georgia, June 16–19, 2003. Responsible person, Peter L. Meitner, organization code 0300, 216–433–3715.

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category: 07 Distribution: Nonstandard

Available electronically at http://gltrs.grc.nasa.gov

This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

A calculation procedure is presented which allows the one-dimensional determination of flow distributions in arbitrarily connected (branching) flow passages having multiple inlets and exits. The procedure uses an adaptation of the finite element technique, iteratively coupled with an accurate one-dimensional flow solver. The procedure eliminates the usual restrictions inherent with finite element flow calculations. Unlike existing one-dimensional methods, which require simplifications to the flow equations (uncoupling the momentum and energy equations), to allow for arbitrary branching and multiple inlets and exits, the only limitation of the described methodology is that, at present, it can only accommodate non-rotating configurations (no pumping effects). The calculation procedure is robust, and will always converge for physically possible flow. The procedure is described, and its use is illustrated by an example.

14. SUBJECT TERMS

Compressible fluid flow; Turbine cooling; Heat transfer

R OF PAGES

11

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	

NASA/TM—2004-211885

ARL–TR–3213
GT2003–38061

Procedure for Determining One-Dimensional Flow Distributions in Arbitrarily Connected Passages Without the Influence of Pumping

Peter L. Meitner
U.S. Army Research Laboratory, Glenn Research Center, Cleveland, Ohio

June 2004

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301–621–0134

- Telephone the NASA Access Help Desk at 301–621–0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/TM—2004-211885

ARL–TR–3213
GT2003–38061

Procedure for Determining One-Dimensional Flow Distributions in Arbitrarily Connected Passages Without the Influence of Pumping

Peter L. Meitner
U.S. Army Research Laboratory, Glenn Research Center, Cleveland, Ohio

Prepared for the
Turbo Expo 2003
cosponsored by the American Society of Mechanical Engineers
and the International Gas Turbine Institute
Atlanta, Georgia, June 16–19, 2003

National Aeronautics and
Space Administration

Glenn Research Center

June 2004

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at http://gltrs.grc.nasa.gov

PROCEDURE FOR DETERMINING ONE-DIMENSIONAL FLOW DISTRIBUTIONS IN ARBITRARILY CONNECTED PASSAGES WITHOUT THE INFLUENCE OF PUMPING

Peter L. Meitner
U.S. Army Research Laboratory
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

ABSTRACT

A calculation procedure is presented which allows the one-dimensional determination of flow distributions in arbitrarily connected (branching) flow passages having multiple inlets and exits. The procedure uses an adaptation of the finite element technique, iteratively coupled with an accurate one-dimensional flow solver. The procedure eliminates the usual restrictions inherent with finite element flow calculations. Unlike existing one-dimensional methods, which require simplifications to the flow equations (uncoupling the momentum and energy equations), to allow for arbitrary branching and multiple inlets and exits, the only limitation of the described methodology is that, at present, it can only accommodate non-rotating configurations (no pumping effects). The calculation procedure is robust, and will always converge for physically possible flow. The procedure is described, and its use is illustrated by an example.

INTRODUCTION

The design of cooling passages in turbomachinery components (vanes and blades) continues to be a challenging undertaking, since the number of possible cooling configurations is virtually infinite. While a number of capable flow/heat-transfer solvers are available [e.g., ref. 1 to 3], they are, for the most part, multi-dimensional final design tools that require substantial, time consuming, detailed geometric inputs, as well as significant run times. A need exists for a simple, one-dimensional "screening" code that can be used to narrow down the number of possible configurations in a timely and cost effective manner.

Reference 4 describes a one-dimensional computer code (called CPF, Coolant Passage Flow), which was developed because existing, company pro-prietary, one-dimensional codes (that can accommodate arbitrary internal flow branching and multiple inlets and exits), make simplifications to the governing equations (uncoupling the momentum and energy equations), leading to loss of calculation accuracy. CPF makes no simplifying assumptions to the governing one-dimensional flow equations. The increased calculation accuracy, however, comes at a price—CPF can only analyze a coolant flow path geometry that consists of a single flow passage with one inlet and one exit.

This paper describes a methodology that couples CPF with a finite element technique flow solver and extends CPF's applicability to arbitrarily connected flow passages having multiple inlets and exits. The main attributes of CPF are retained, except for the ability to account for pumping (rotational effects). The described methodology is, for now, applicable only to cooling geometries in non-rotating components. The described methodology is not limited to CPF, but can be implemented with any one-dimensional flow/heat-transfer code.

CPF (COOLANT PASSAGE FLOW)

The computer code CPF is described in ref. 4, which is available in the open literature. CPF was developed specifically for radial turbomachinery, but can be used to analyze any coolant flow path geometry that consists of a single flow passage with one inlet and one exit. The flow path is defined by nodes (using Cartesian coordinates at the passage centroids) and by intervals, also referred to as elements. Figures 1 and 2 [from ref. 4] illustrate the nodal/element representation of a typical coolant flow passage for a hypothetical, cooled, radial turbine. Note that CPF can also accommodate tip-cap impingement cooling, and that a flow bypass can be specified, in which coolant flow is taken off at one point in the flow channel and "reintroduced" at a point further downstream in the channel (as shown in fig. 1 and 2). These additional features, however, are not used in the described, coupled calculation procedure.

CPF predicts the coolant mass flow rate, velocity, pressure, and temperature, as well as the coolant heat transfer coefficient distribution along the passage flow path from inlet to exit (for specified inlet and exit pressures and for specified or empirical friction and heat transfer correlations). The solution is limited to subsonic flow. From ref. 4, the coupled, one-dimensional momentum and energy equations (without the pumping terms and the mass "deletion/reintroduction" terms) express the rates of change of pressure and temperature with path length (x) along an element as follows:

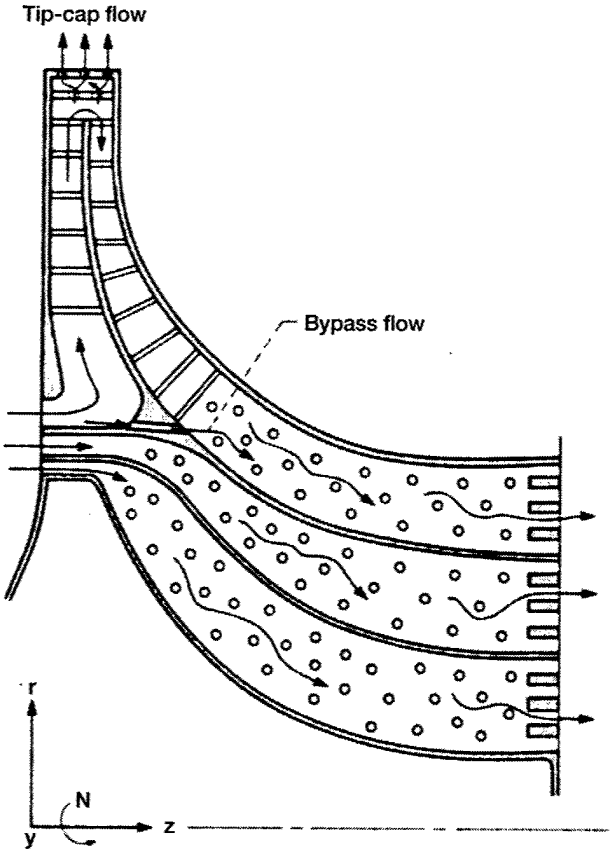

Figure 1.—Conceptual cooled radial turbine configuration (from ref. 4).

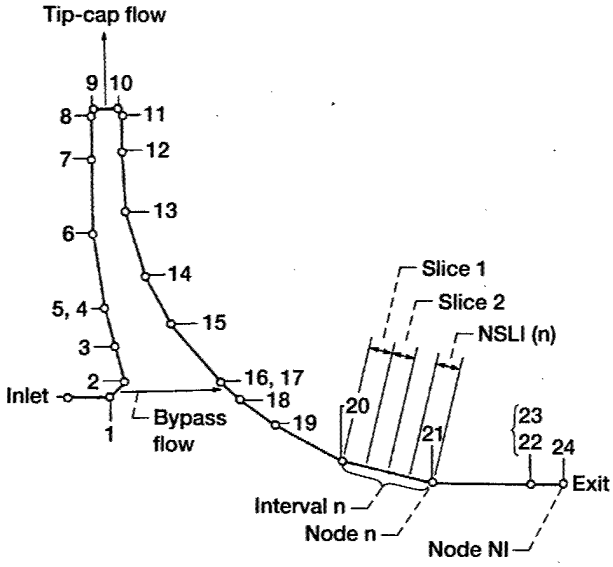

Figure 2.—Nodal representation of figure 1 upper flow path (from ref. 4).

$$\frac{dp}{dx}\left(1-\frac{(\dot{w}/A)^2 Rg_c T}{p^2}\right)=\frac{(\dot{w}/A)^2 Rg_c T}{pA}\frac{dA}{dx} - \frac{(\dot{w}/A)^2 Rg_c}{p}\frac{dT}{dx} - \frac{4fRg_c T(\dot{w}/A)^2}{2pD_h} \quad (1)$$

and

$$\frac{dT}{dx}\left(\dot{w}C_p+\frac{\dot{w}^2 Rg_c T}{pA}\frac{\dot{w}Rg_c}{pA}\right)=\left(\frac{\dot{w}^2 Rg_c T}{pA}\frac{\dot{w}Rg_c T}{pA^2}\right)\frac{dA}{dx} + \left(\frac{\dot{w}^2 Rg_c T}{pA}\frac{\dot{w}Rg_c T}{p^2 A}\right)\frac{dp}{dx} + h_c P(T_w - T_{aw}) \quad (2)$$

where

A	cross sectional area
C_p	specific heat at constant pressure
D_h	hydraulic diameter
f	Fanning friction factor
g_c	gravitational constant
h_c	heat transfer coefficient
p	static pressure
P	perimeter
R	universal gas constant
T	temperature
T_w	wall temperature
T_{aw}	adiabatic wall temperature
x	path length
$\dot{w}$	mass flow rate

CPF integrates these coupled equations (1 and 2) along the flow path. The above equations account for friction, area change, and heat transfer. Friction factors and heat transfer coefficients for each element are obtained from empirically derived correlations.

THE FINITE ELEMENT METHOD

The finite element method for fluid flow calculations is described in ref. 5. Flow passages are divided into elements that are connected at junction points called nodes. A simple stick diagram flow network (for this paper's example problem) is shown in fig. 3, overlaid on a z-r Cartesian grid. The numbers in square boxes designate the elements, while those within parentheses designate the node points (shown as solid circles). Note that, for this example, the number of nodes (eight) is equal to the number of elements, which is usually not the case. Also note that any number of elements can be connected to a single node. Flow directions are assumed (arrows next to element numbers) in each element based upon known pressures at the entrance and exit nodes. In fig. 3, nodes 1, 2, and 5, are entrance nodes, and nodes 7 and 8 are exit nodes. Nodes 3, 4, and 6 are internal nodes, where the pressures are unknown and must be predicted. The diagram in fig. 3 is an arbitrary connection of flow passages and does not represent any real or practical network.

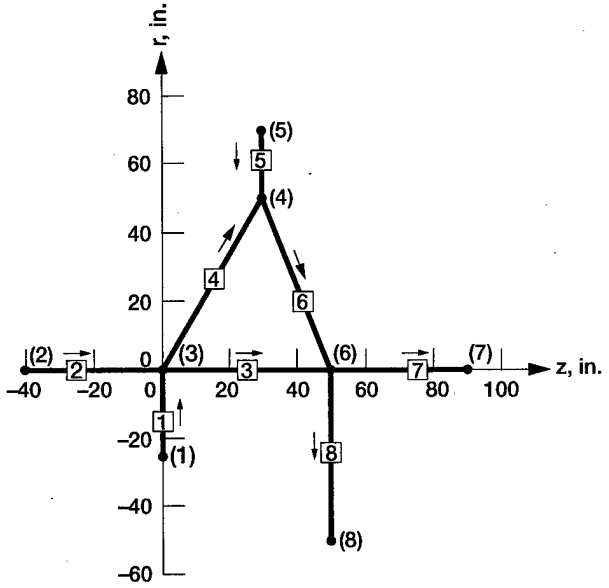

Figure 3.—Example problem stick diagram flow network.

In the finite element flow calculation method, the volume flow rate (Q) for each element is expressed as the product of a "stiffness coefficient", or "flow function", (K) times the change in pressure across the element.

$$Q = -K \bullet \Delta p \tag{3}$$

where

$$K = (\pi \bullet D^4)/(k \bullet \mu \bullet L) \tag{4}$$

and

D	element equivalent pipe diameter
k	a constant (dependent on the physical units being used)
L	element length
Δp	pressure drop across the element
μ	fluid dynamic viscosity

The above expression for Q forces serious limitations on the types of allowable flow (discussed below), but results in a fixed value of K for each element. The individual K's are incorporated into an overall, global "stiffness matrix", which is solved (by matrix techniques) in a single step to give volume flow rates for all elements, as well as the previously unknown pressures at the internal node points. If the assumed flow directions are correct, the calculated volume flows are positive. If the flow direction in an element is opposite to the initial guess, the calculated volume flow is a negative number.

At first glance, the finite element method would appear to be an unlikely candidate for flow and heat transfer calculations, since, in its most fundamental form, it suffers from the following, serious limitations: (1) the fluid must be incompressible; (2) the fluid flow must be laminar; (3) each passage element must be of constant cross sectional area; (4) no allowance is made for temperature variations (heat transfer); and (5) no differentiation is made between total and static pressures. However, as will be shown, all these limitations are bypassed by incorporating flow information generated by CPF. Only the finite element solution method is utilized in the described, coupled, iterative solution procedure.

COUPLING OF CPF AND THE FINITE ELEMENT METHOD

To couple CPF and finite element flow calculations, it is assumed that the "real world" mass flows through each element of a multiple flow path can also be expressed (as in the finite element method) by the product of a "flow function" and the pressure drop across the element. The "real world" flow functions must now, however, encompass all possible types of flow (incompressible or compressible, laminar, or turbulent), as well as allow for varying flow area, friction, and heat transfer within the elements. The values of the individual flow functions are, of course, unknown, and are obtained (using CPF) in the following, iterative manner, as diagrammed in fig. 4.

From the flow passage diagram (e.g., fig. 3) and the known entrance and exit pressures, flow directions are assumed (arrows next to the element numbers in fig. 3). Next, an initial guess is made for the unknown node pressures at the internal node points, consistent with the assumed flow directions and the specified entrance and exit pressures. Then, for each element, CPF is used to calculate the flow rate and the "flow function" (K) by dividing the calculated flow rate by the initially assumed pressure drop. Finally, the calculated flow functions are input into the finite element flow solver, which predicts updated pressures for all the internal nodes.

These updated internal node pressures are then again used by CPF to calculate further updated flow rates and flow functions for use by the finite element method to obtain the next set of updated internal node

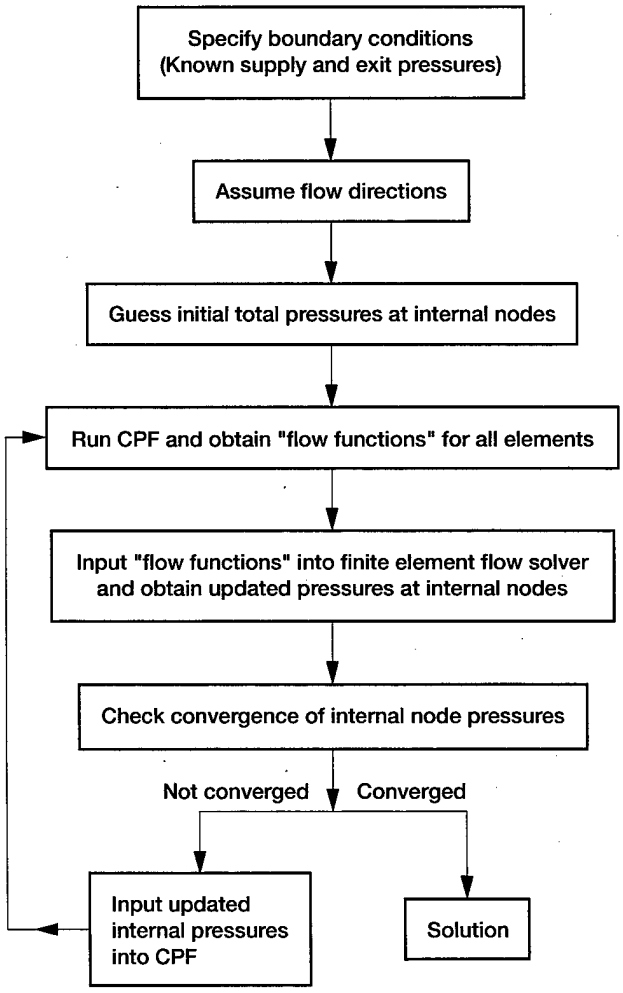

Figure 4.—Iterative flow calculation diagram.

pressures. The procedure is repeated until convergence is achieved within a specified tolerance on successively calculated pressures at all internal nodes. At that point the sum of all inlet flows will equal the sum of all exit flows.

As stated previously, in CPF, empirically derived correlations are specified for friction factor and heat transfer coefficients. These correlations can be specified in a number of ways, including from built-in correlations or from curves input in tabular form. This feature, (specifying friction factor and heat transfer coefficients in a multitude of ways) is not yet available in the prescribed, coupled solution procedure. Friction factor and heat transfer coefficients must, for now, be specified directly by the user for each interval.

OTHER COUPLING CONSIDERATIONS

The preceding, simple description of the coupled solution procedure does not address the myriad of details that are necessary to assure a correct and meaningful flow solution. Only a few of the other considerations can be addressed here due to paper length limitations. Some of the most important are as follows:

Pressure. As stated previously, in the finite element flow calculation method, no differentiation is made between total and static pressures. The procedure, however, requires decreasing pressure along all element flow paths. Therefore, all pressure information passed between CPF and the finite element flow solver must be based on total conditions. While static pressure varies significantly with flow area, total pressure remains constant, unless varied by friction and/or heat transfer. For the types of fluid flow problems encountered by turbomachinery designers, friction will always play a significant role in reducing total pressure along the flow path.

In CPF, total pressure is specified at the flow path inlet, and the corresponding inlet static conditions are determined from the calculated inlet velocity. Accordingly, for the described coupled solution method, the known supply pressures at inlet nodes (entrances to elements), as well as the initial guesses for the unknown pressures at the internal nodes are specified as total pressures. The known exit pressures are specified as static pressures, but only the corresponding, calculated total pressures at the exit nodes are passed on to the finite element flow solver. Also, for each element, an inlet total pressure loss coefficient is specified.

The finite element method requirement for decreasing pressure in each element is the reason why the described, coupled solution procedure does not work in a rotating environment. Pumping (caused by rotation, coupled with increasing path radius) increases total pressure along the passage, and can easily overcome the total pressure drop due to friction. Efforts to overcome this limitation have so far involved modifying the stiffness coefficient(s) in the appropriate element(s). While, at times, this has allowed some additional iteration steps to be completed, no overall converged solution procedure has yet been achieved in the presence of pumping.

Temperature. In the described coupled solution method, total temperatures are also specified at all inlets and as initial guesses at the internal nodes. If two or more elements feed flow into an internal node, a "mass averaged" node total temperature is calculated. This average total temperature is then used as the inlet total temperature for those elements flowing out of the internal node.

Flow Direction. As stated previously, if, in the finite element method, the initial guess of flow direction in any element is wrong, the predicted volume flow rate will be a negative number (although still correct), indicating that the flow is opposite to the initially assumed direction. This is not the case with the coupled solution method. If an initially assumed flow direction is wrong, CPF will fail to converge for that particular element, since, without pumping, the flow is only allowed to travel in the direction of decreasing total pressure. If this happens, appropriate error messages identify the element, and the user must change the input to incorporate the opposite element flow direction.

SAMPLE CALCULATION

A sample calculation was performed, using English units, for a flow network based on the stick diagram of fig. 3. For simplicity, the example was run with fixed physical gas properties (gamma = 1.4, C_p = 0.24 BTU/lbm/R, viscosity = 1.21 $\times 10^{-5}$ lbm/ft/s, gas constant = 53.35 ft•lbf/lbm/R). Zero heat transfer and zero inlet losses were assumed, and the Fanning friction factors were 0.20 for all elements. Nodes 1, 2, and 5 are inlets, nodes 7 and 8 are outlets, and nodes 3, 4, and 6 are internal nodes where the unknown pressures have to be determined. Elements 1 and 3 have constant cross sectional area, while all other elements have varying cross sectional areas. Inlet total pressures at nodes 1, 2, and 5 were specified as 50.0 psia each, while inlet total temperatures at those nodes were specified as 500.0, 400.0, and 300.0 °F, respectively. Exit static pressures at nodes 7 and 8 were specified as 20.0 and 15.0 psia, respectively. The initial guesses for total pressures at nodes 3, 4, and 6 were 47.0, 46.0, and 45.0 psia, respectively. Table 1 shows the pertinent input information for each element in tabular form.

TABLE 1.—EXAMPLE PROBLEM ELEMENT TABULATION OF PRESCRIBED PARAMETERS

Element, [n]	Inlet node, (n)	Outlet node, (n)	Inlet z, in.	Outlet z, in.	Inlet r, in.	Outlet r, in.	Inlet area, in.^{2}	Outlet area, in.^{2}	Inlet total pressure, psia	Outlet static pressure, psia	Inlet total temperature, °F
1	1	3	0.0	0.0	–25.0	0.0	2.0	2.0	50.0	-----	500.0
2	2	3	–40.0	0.0	0.0	0.0	2.0	3.0	50.0	-----	400.0
3	3	6	0.0	50.0	0.0	0.0	3.0	3.0	-----	-----	-------
4	3	4	0.0	30.0	0.0	50.0	2.0	1.0	-----	-----	-------
5	5	4	30.0	30.0	70.0	50.0	2.0	1.0	50.0	-----	300.0
6	4	6	30.0	50.0	50.0	0.0	2.0	1.0	-----	-----	-------
7	6	7	50.0	90.0	0.0	0.0	2.0	3.0	-----	20.0	-------
8	6	8	50.0	50.0	0.0	–50.0	2.0	1.0	-----	15.0	-------

TABLE 2.—CALCULATED ELEMENT PARAMETERS
(Prescribed parameters in parentheses)

Element, [n]	Flow rate, lbm/s	Inlet total pressure, psia	Outlet total pressure, psia	Inlet static pressure, psia	Outlet static pressure, psia	Inlet total temperature, °F	Outlet total temperature, °F	Inlet static temperature, °F	Outlet static temperature, °F	Inlet, M	Outlet, M
1	0.2349	(50.0)	47.14	49.78	46.91	(500.0)	500.0	498.8	498.6	0.0795	0.0844
2	0.2503	(50.0)	47.14	49.78	47.04	(400.0)	400.0	398.9	399.5	0.0802	0.0566
3	0.4771	47.144	37.56	46.74	37.05	448.4	448.4	446.2	444.8	0.1115	0.1405
4	0.0068	47.144	47.13	47.14	47.13	448.4	448.4	448.4	448.4	0.0024	0.0048
5	0.1884	(50.0)	47.13	49.89	46.65	(300.0)	300.0	299.5	297.8	0.0566	0.1210
6	0.1946	47.130	37.56	47.00	36.91	305.2	305.2	304.6	301.4	0.0623	0.1583
7	0.4567	37.560	20.84	36.53	(20.0)	406.9	406.9	400.1	396.8	0.1995	0.2425
8	0.2163	37.560	17.26	37.34	(15.0)	406.9	405.5	405.4	371.4	0.0927	0.4526

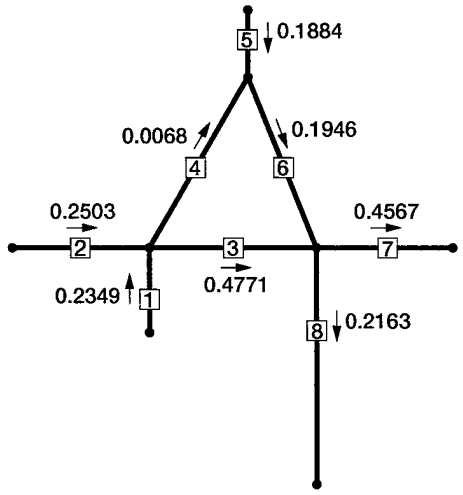

Figure 5.—Calculated flow rates (lbm/s).

RESULTS

The solution procedure converged in 10 overall iterations, requiring less than 10 seconds of computer time, using a 550 MHz personal computer. The calculated flow rates are shown in fig. 5, while table 2 tabulates further calculated flow information for each element. From fig. 5 it can be seen that the sums of the calculated flow rates going into and out of the internal nodes (3, 4, and 6), as well as the overall inlet and exit flows (0.6736 and 0.6730 lbm/s, respectively), agree within the 3rd significant figure. This is reassuring, since the convergence criterion is not based on flow rates, but on the relative differences between successively predicted pressures at the interior node points. The convergence criteria is that, for each internal node, ABS(PTOLD-PTNEW)/PTOLD is less than or equal to 5×10^{-4}, where PTOLD and PTNEW are the previous and latest calculated node total pressure, respectively. The calculation procedure also converged for other initial pressure guesses at the internal node points (as long as they did not violate the assumed flow direction criteria), showing that the overall iteration procedure is robust.

CONCLUSIONS

1. An iterative calculation procedure has been developed that couples the finite element flow solution method with an accurate one-dimensional flow and heat transfer solver called CPF.
2. The described procedure is shown to be capable of predicting the one-dimensional fluid flow distribution in arbitrarily connected passages having multiple inlets and exits by simultaneously solving the coupled one-dimensional momentum and energy equations. The procedure differs from other, known methods, which require simplifications (uncoupling) of the governing equations.
3. The calculation procedure can be adapted to any one-dimensional flow solver, and can accommodate any fluid flow that is consistent with the one-dimensional flow code (e.g. compressible, incompressible, laminar or turbulent flow with varying flow area, friction, and heat transfer).
4. The solution procedure is robust, and has always converged rapidly for physically possible flow.
5. At present, the only limitation of the calculation technique is the inability to accommodate pumping effects. Efforts to overcome this limitation are in progress.

REFERENCES:

1. Steinthorsson, E., Liou, M.S., and Povinelli, L.A., "Development of an Explicit Multiblock/Multigrid Flow Solver for Viscous Flows in Complex Geometries", AIAA–93–2380, 1993. Also: NASA TM–106356 and ICOMP 93–34.
2. Steinthorsson, E., Ameri, A.A., and Rigby, D.L., "Simulations of Turbine Cooling Flows Using a Multiblock-Multigrid Scheme", NASA CR–198539, 1996. Also AIAA–96–0621 and ICOMP 96–7.
3. Heidmann, J.D., Rigby, A.L., and Ameri, A.A., "A Three-Dimensional Coupled Internal/External Simulation of a Film-Cooled Turbine Vane", ASME 99–GT–186, 2000.
4. Meitner, P.L., "Computer Code for Predicting Coolant Flow and Heat Transfer in Turbomachinery", NASA TP–2985, AVSCOM TP 89–C–008, 1990.
5. Allaire, P.E., "Basics of the Finite Element Method—Solid Mechanics, Heat Transfer, and Fluid Mechanics", W.C. Brown Publishers, 1985.